Machining
ALCOA ALUMINUM

1943

Machining Alcoa Aluminum

AND ITS ALLOYS

ALCOA
ALUMINUM

ALUMINUM COMPANY
OF AMERICA

PITTSBURGH · PENNSYLVANIA

1943

CONTENTS

(Continued on next page)

TABLES

MOTION PICTURE ON MACHINING ALUMINUM

"How to Machine Aluminum", a black-and-white sound motion picture outlines the best practices for machining aluminum with either hand or machine tools. 32 minutes.

The picture may be borrowed from the Motion Picture Department, Aluminum Company of America, Pittsburgh, Pa., without charge other than transportation costs. Prints are available in both 16 mm. and 35 mm. widths. Please specify the width, and give a first and second choice of dates when ordering.

This is one of the Alcoa how-to-do-it motion pictures. Other pictures in the series include "How to Weld Aluminum" and "How to Rivet Aluminum."

INTRODUCTION

From this booklet the machinist can gain an understanding of the characteristics peculiar to aluminum and its alloys, as distinguished from those of other commonly used metals, as well as a knowledge of the machining practices best adapted for it.

As commonly used tools for cutting steel will often perform satisfactorily on aluminum, a machinist will be able to machine aluminum and its alloys successfully by supplementing his experience with the information contained herein.

It is the purpose of this booklet: (1) To set forth the general principles of machining aluminum and its alloys; (2) to suggest speeds, feeds, and depths of cut which will produce satisfactory results; (3) to point out where common practice as well as tools of standard design may be used; and (4) to indicate where the use of special practices or tools is desirable.

Part 1 deals with general machine shop practice. Part 2 describes the practices employed in automatic screw machine operations.

1

General Machining Practice

ALUMINUM ALLOYS

THE word "aluminum" is commonly used to describe the pure metal and all its alloys, in either the cast or wrought condition. Some of the alloys have better machining properties than others, in that they can be cut fast, the chips are small, and smooth surfaces can be produced readily. Others are more difficult to machine—they produce cuttings that are long and stringy, while still others are soft and gummy.

In Table 1, page 41, commercial aluminum alloys are segregated into groups depending on whether they are cast or wrought, heat treated or not heat treated. Within each group the alloys are arranged in the approximate order of their relative machinability. The alloys in Group I have the best machinability, those in Group II the next, and those in Group III the poorest.

The casting alloys containing principally copper, magnesium, or zinc can generally be machined most rapidly and satisfactorily. The tools may have smaller rake angles than those required for most of the other alloys, the chips are small, and there is little or no tendency for the tools to leave a burr or for chips to build up on the cutting edge. On the other hand, the casting alloys in which silicon is the predominant alloying element machine best if the speeds and cuts are reduced and the rake angles increased. The alloys containing relatively large amounts of silicon are abrasive to carbon and high-speed steel tools and should be machined with cemented carbide tipped tools.

Of the wrought alloys, those which depend on various amounts of work hardening to improve their mechanical properties are easy to machine with tools having relatively large rake angles. Other alloys, which may be heat treated to improve their mechanical

properties, also generally have good machining characteristics. Among the latter are 17S-T, 24S-T, and 53S-T alloys which are widely used for automatic screw machine applications. One commercial alloy in particular, 11S-T3, is free-cutting to a high degree, having been developed especially for screw machine work; it can be machined readily at high speeds and heavy feeds, and the chips are small.

Even the softest aluminum, including high purity metal in the annealed temper, may be machined with excellent results when large rake angles are employed and the tools are carefully finished with a fine abrasive stone.

PRINCIPAL CHARACTERISTICS OF TOOLS FOR ALUMINUM

Although the machining properties of the various alloys differ as indicated above, the following practices in general characterize the principal differences between aluminum and most other metals with respect to the tools required and should therefore be carefully observed:

1. Grind more top and side rake on the cutting tools than is common for machining steel.
2. Keep cutting edges sharp and free of burred or wire edges.
3. Maintain smooth, bright tool surfaces free from scratches.

TOOL MATERIALS

Tools of plain high-carbon steels frequently perform satisfactorily when machining aluminum and most of its alloys. Under conditions in which the cutting speeds are necessarily low, they may be the most economical and this is particularly true for small diameter drills. For quantity production work, tools made from high-speed steels have largely replaced the carbon steel tools, but in many instances, tools tipped with cemented carbides have proved far superior to high-speed steel tools. Tools of this last type are especially suited for the machining of aluminum alloys of high silicon content; in fact some of these alloys cannot be machined

successfully under production conditions without it. The cemented carbide tipped tools, when ground to the rake angles suggested in this booklet, produce excellent machined surfaces and remain sharp for long periods of time without regrinding; consequently these tools are economical for high rate production. The use of cemented carbide tools is, of course, restricted to operations in which the work is free from vibration and irregularities in the cut.

TOOL SHAPES IN GENERAL

In the following description of tool shapes, a wide range of rake angles is indicated. In general, the larger rake angles are employed for finishing tools and for the aluminum alloys that are not free-cutting; this includes the softer materials which require tools with exceptionally acute and keen cutting edges. On the other hand, rake angles in the lower range are used for roughing cuts and for machining the alloys that have free-cutting characteristics. Tools similar to those used for machining steel may often be employed successfully.

Top Rake—Top rake, frequently called "hook" because it gives a hook-like appearance to the tool, generally varies from 20 to 50 degrees. Very finely finished surfaces may be produced with tools having a top rake angle in the higher end of the range, but, obviously, such a tool can be used only in a machine that is sturdy, free from vibration, and which has no lost motion in the feeding mechanism. For some operations it may be necessary to use a top rake smaller than indicated by the above range, but a negative rake should never be used.

Side Rake—Side rake is important in machining aluminum and its alloys as this produces a slicing action which is especially effective in parting the cutting from the stock. A side rake of from 10 to 20 degrees assists materially in the cutting action of the tool. Planer and shaper tools may have a considerable amount of side rake; finishing tools have been ground with a side rake as high as 60 degrees in order to take full advantage of the free-cutting characteristics of aluminum.

Clearance—The clearance should be about 8 to 10 degrees. The clearance must be carried around the side of the tool which advances into the work. This angle is important. If too small, the side of the tool will rub against the work and generate heat. If too large, the tool may tend to dig into the work or chatter.

Tool Finishes—In all cases it is essential that the cutting edges be keen, smooth, and free from grinding wheel scratches, burrs, or wire edges. Too much emphasis cannot be given to tool finish, because on it depends, to a large extent, the success of machining aluminum and its alloys. Keen edges are best obtained by finish grinding on a fine or very fine abrasive wheel, then hand stoning with a fine or very fine oilstone, or lapping, taking care that neither the angles nor the contour of the cutting edge are appreciably

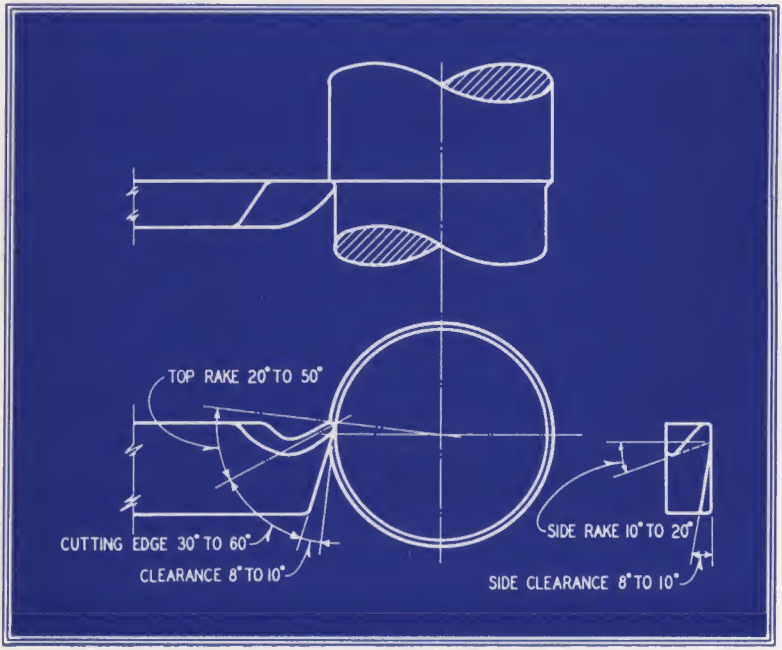


Figure 1 — *Lathe tool is set at or slightly above center.*

Figure 2 — *Lathe tool and holder.*

modified during the finishing operations. Where possible, cemented carbide tools should be diamond lapped.

ENGINE LATHE PRACTICE

Turning Tools—For ordinary engine lathe work a round-nosed tool, as shown in Figure 1, may be used. General practice is to set the tool at or slightly above center. Sturdy construction of tools and holders is essential to minimize vibration at the high speeds at which aluminum alloys are machined. Although the same tool may often be used for both roughing and finishing cuts, it is important that the cutting edge be restoned before the finishing operation.

Figure 1 indicates the conventional type of solid lathe tool made from rectangular stock. The tool bits used in some of the patented lathe tool holders may likewise be ground in accordance with the angles shown on this figure. Figure 2 illustrates such a tool and holder.

Another type of tool and holder which possesses certain adjustable features is shown in Figure 3. The bit of this tool is made from round rod stock of high-carbon or high-speed steel, properly hardened and tempered. Resharpening is readily accomplished by

holding the bit by its shank in the chuck or collet of a tool grinding machine or an engine lathe, and grinding off the outside diameter until a keen edge is obtained. After each grinding, the tool should be stoned on the top surface. By using such a tool and following the suggested resharpening procedure, the desired shape may be easily maintained. When the clamp-screw of the tool bit holder is loosened the bit may be turned to various positions, making it adjustable to different working conditions. Tools of this form may be used for both rough turning and finishing cuts.

Parting Tools—Parting tools for machining aluminum and its alloys should have from 12 to 20 degrees top rake and should be stoned so that their cutting edges are keen and smooth. With such tools, the clearance angle should be only about 3 or 4 degrees and light feeds should be used.

Boring Tools—In general, the angles indicated in Figure 1 should be employed for boring tools, except that the clearance angle must be larger for small bores; otherwise, the lower portion of the tool

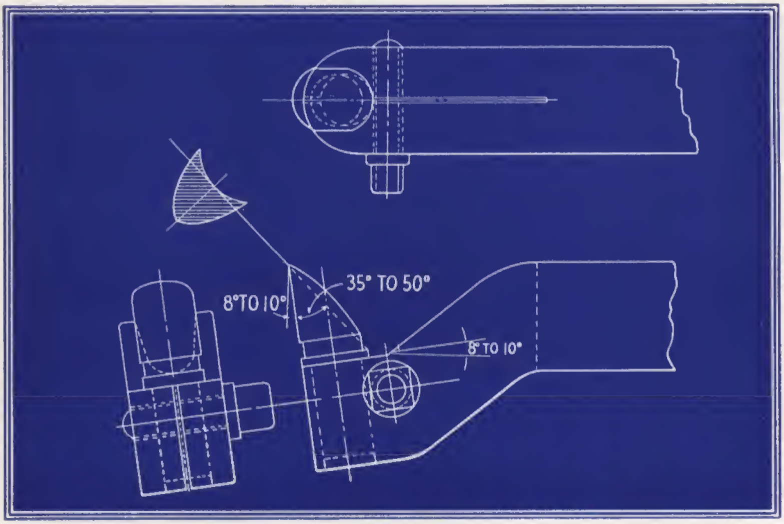


Figure 3 — *Lathe tool with bit that is easily ground to the proper shape.*

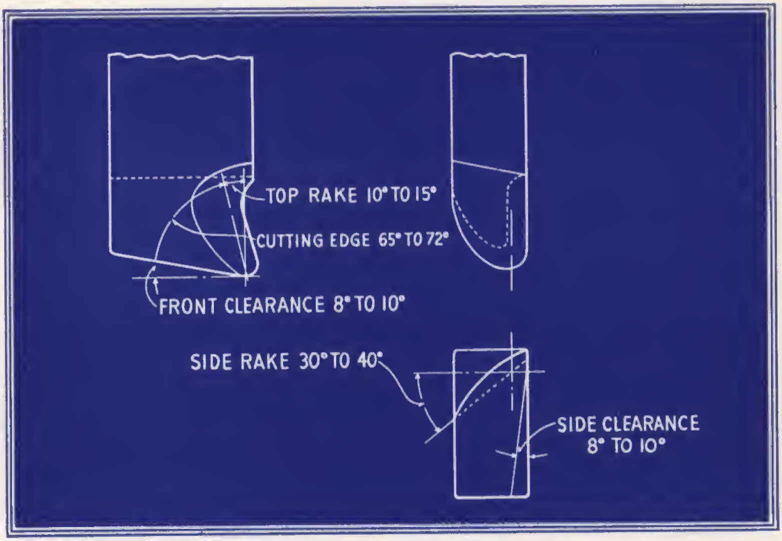


Figure 4 — *Planer tool for roughing cuts.*

may rub the work and prevent the tool from cutting. By proper design of the tool and holder, the tool shown in Figure 3 may also be adapted to boring operations in an engine lathe or a boring machine.

Cuttings—When turning some of the aluminum alloys using a tool with considerable rake, the cuttings may be continuous and slightly curled. Decreasing the rake angles may tend to curl the cuttings more and cause them to break up. The extent to which this may be done and yet obtain the desired surface finish depends largely upon the alloy being machined. Long continuous cuttings may be objectionable for two reasons; they may foul the tool and machine, and they may also rub over the finished portion of the work and scratch the surface, since they are harder than the stock because of the cold working they have received during the cutting operation.

Lathe Centers—In engine lathe work, when using heavy, coarse-feed cuts with aluminum, excessive friction is encountered on the lathe centers because of the expansion of the metal with rise in

temperature. Ball or roller bearing tail-stock lathe centers assist in decreasing this trouble.

Cuts, Feeds and Speeds—Turning speeds, feeds and cuts for aluminum alloys are shown in Table 2, page 42.

PLANER AND SHAPER TOOLS

The inertia of the table and ram limit the speed at which aluminum alloys can be cut in a planer or shaper. As the work can generally be anchored securely, heavy feeds and cuts can be taken, which, to some extent, compensate for the low attainable cutting speeds.

A roughing tool is shown in Figure 4. It is a sturdy tool with only a moderate amount of rake.

A finishing tool is shown in Figure 5. This has considerable top rake and an extremely large amount of side rake. This gives a long sweep of the cutting edge and produces a decided slicing action which cuts aluminum freely. This tool should be used for light cuts

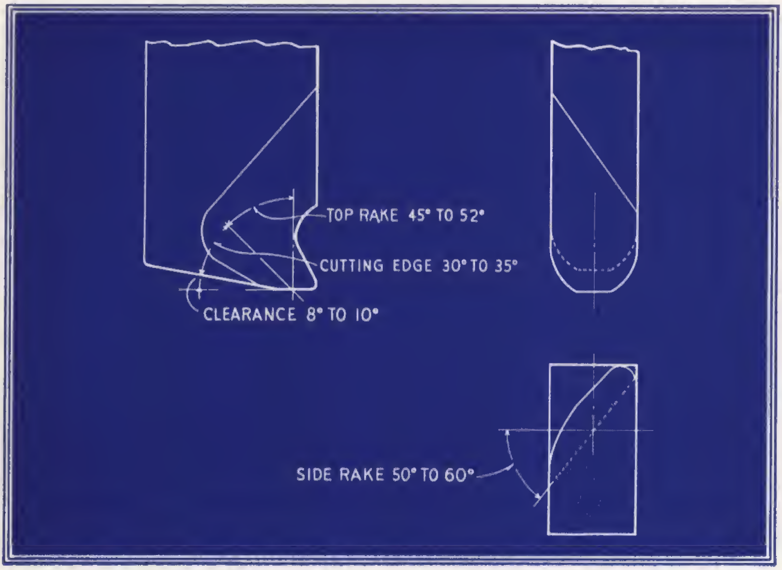


Figure 5 — *Planer tool for finishing cuts.*

with fine feeds only. Care should be taken to prevent the tool from striking the work on its return stroke. Observance of this precaution keeps the finished work from being marred and also prevents the thin edge of the tool from being injured.

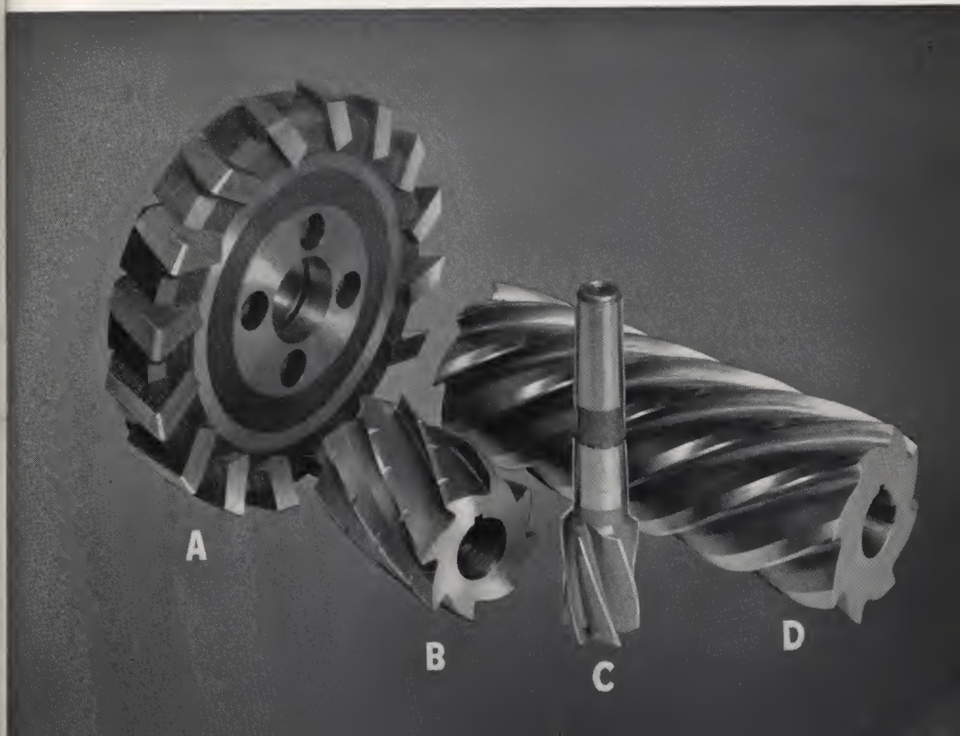
Feeds and cuts at the maximum speed of the equipment are suggested in Table 2, page 42.

MILLING CUTTERS

All types of milling cutters for machining aluminum should correspond in so far as practical to the general tool shapes described above. They should have coarse teeth and, where possible, the teeth should be designed to afford top and side rake. Nicked teeth break the cuttings into smaller pieces and are helpful in some instances, but they are not generally essential for good results. Milling cutters for aluminum are illustrated in Figure 6.

For the plain type milling cutter, the coarse-tooth, heavy-duty

Figure 6 — (A) *Inserted-tooth face milling cutter;* (B) *spiral nicked-tooth plain milling cutter;* (C) *end mill for milling aluminum; and* (D) *helical milling cutter.*



cutter with spiral teeth is well suited for machining aluminum. A 3-inch diameter cutter with 8 teeth, deeply cut, provides ample clearance for cuttings. The teeth should be undercut to provide a top rake angle of 10 to 20 degrees. These cutters are generally made with a spiral angle of about 25 degrees. The spiral angle produces side rake, giving the slicing action which is so desirable for the efficient cutting of aluminum. The more recent helical milling cutters, primarily designed for machining steel, perform satisfactorily on aluminum and its alloys when the cutting edges are provided with considerable top rake. Some of these are made with a helix angle of about 50 degrees.

The principles outlined above also apply to other types of milling cutters, such as straddle mills, end mills, and face and side milling cutters. These various kinds of cutters are obtainable on the market in the solid or inserted-tooth types.

Aluminum alloys may be milled at relatively high speeds. The best cutting speed, feed, and cut for a job depends on such factors as the type and design of cutter, the kind of tool material used, the sturdiness of the milling machine, its power, and its ability to hold the work securely. The speed at which metal can be removed is limited mostly by the equipment, and extremely high speeds are possible under proper conditions. These possibilities are discussed in the section "Cutting Speeds and Feeds", on page 25.

Speeds, feeds, and cuts for milling aluminum alloys, applicable to average shop conditions, are shown in Table 2, page 42.

THREADING TOOLS

Hand and machine taps of the ground thread type will produce smooth accurate threads in aluminum when they have flutes that are undercut to provide "hook" to the leading edges. The flutes should be deep and wide to provide chip clearance; taps with small flutes are not very satisfactory because chips may pack in the flutes and cause tap breakage or damage to the threads.

Straight-fluted taps are satisfactory for many aluminum alloys, especially those of Types I and II (Table 1, page 41). Spiral-fluted taps like the one illustrated in Figure 7 may be used for any of the

Figure 7 — *Spiral-fluted tap.*

alloys; they are better than straight-fluted taps, especially for tapping soft material. Spiral-fluted taps for cutting right-hand threads should have a right-hand spiral of about the same spiral angle as that used on an ordinary twist drill.

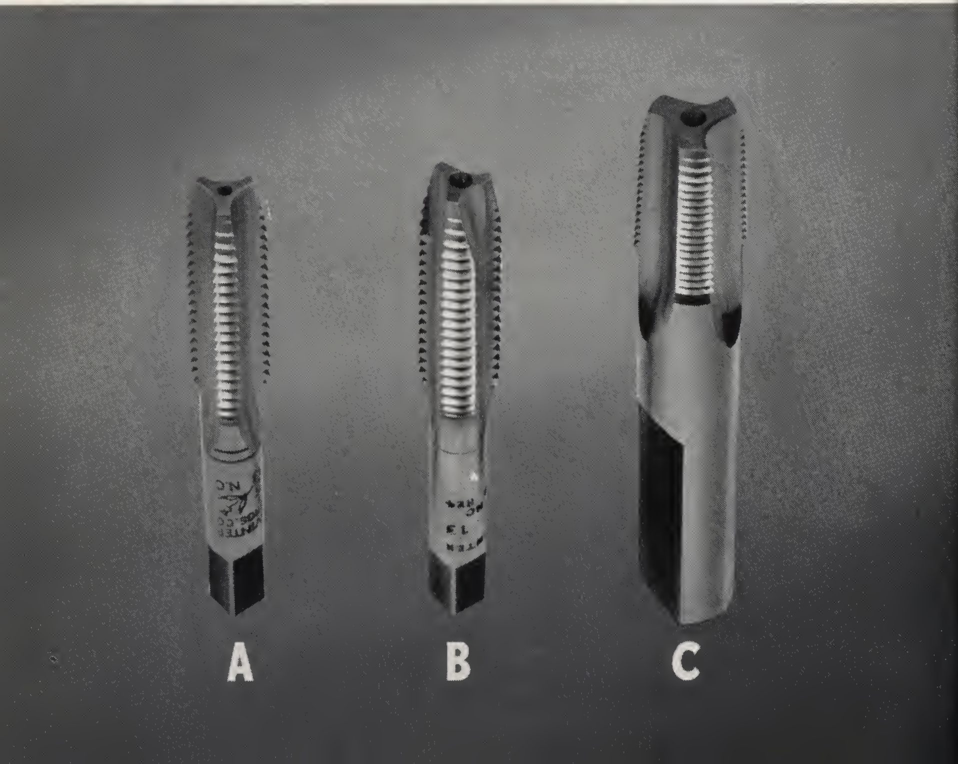
Some taps have a short spiral ground on the front end like the one illustrated in the center of Figure 8. These are generally known

as spiral-pointed or "Gun" taps. This type of tap cuts aluminum alloys freely. Most of the cutting occurs at the end of the tap and the cuttings curl ahead of the tool. It is therefore suited only for operations where there is room for the cuttings to be forced ahead of the tool, as in through holes or blind holes that are deep enough for the chips to collect at the bottom. Taps of this type, however, are not suitable for cutting tapered threads or for use as bottom taps.

Thread chasers for self-opening die heads and collapsible taps should be ground with suitable rakes, clearance and chamfer, as shown in Figure 9. The top rake angle should be in the higher range for machining soft alloys and in the lower range for the harder ones. When selecting this type of equipment, consideration should be given to the disposal of the cuttings, as some tools offer much freer exit for cuttings than others. Provision for lubrication is also important.

Excellent threads may be chased on an engine lathe even in the softest aluminum, by using a single-pointed threading tool. The

Figure 8 — (A) *Standard tap;* (B) *spiral-pointed screw thread tap; and* (C) *pipe thread tap.* (*All with deep flutes.*)



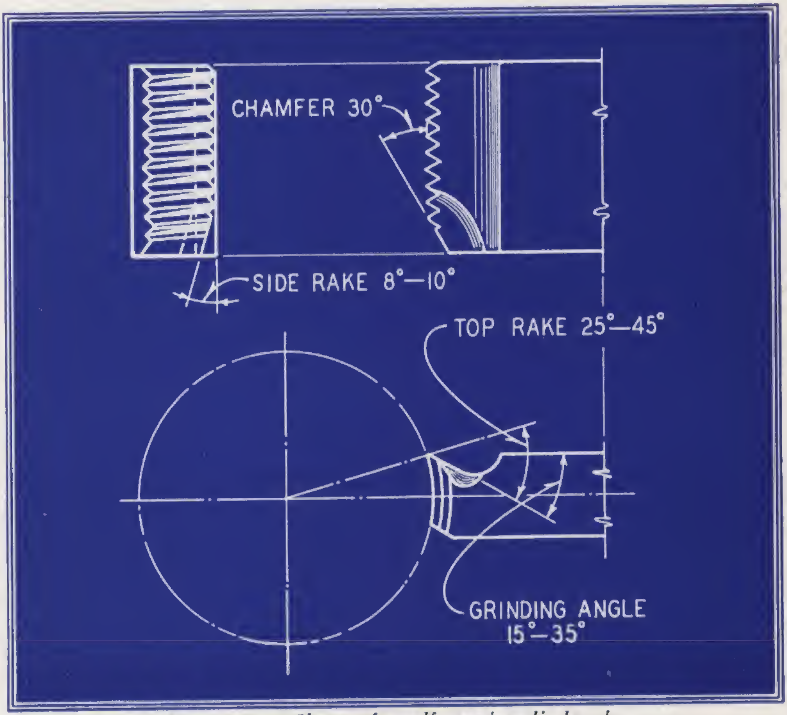


Figure 9 — *Chasers for self-opening die heads.*

top and side rake should be in the lower range indicated above for lathe tools, and the tool should be properly ground to give the required thread contour. This tool is fed into the work at an angle of 30 degrees, using the compound feed.

DRILLS

The standard type twist drill performs satisfactorily on aluminum but better results can be obtained, especially when drilling soft materials, with drills having a larger spiral angle, that is, more twists per inch. The increased spiral gives more "hook" to the cutting edges and causes the drill to cut more freely. It is also helpful in removing cuttings in deep drilling operations. Both kinds of drills are illustrated in Figure 10.

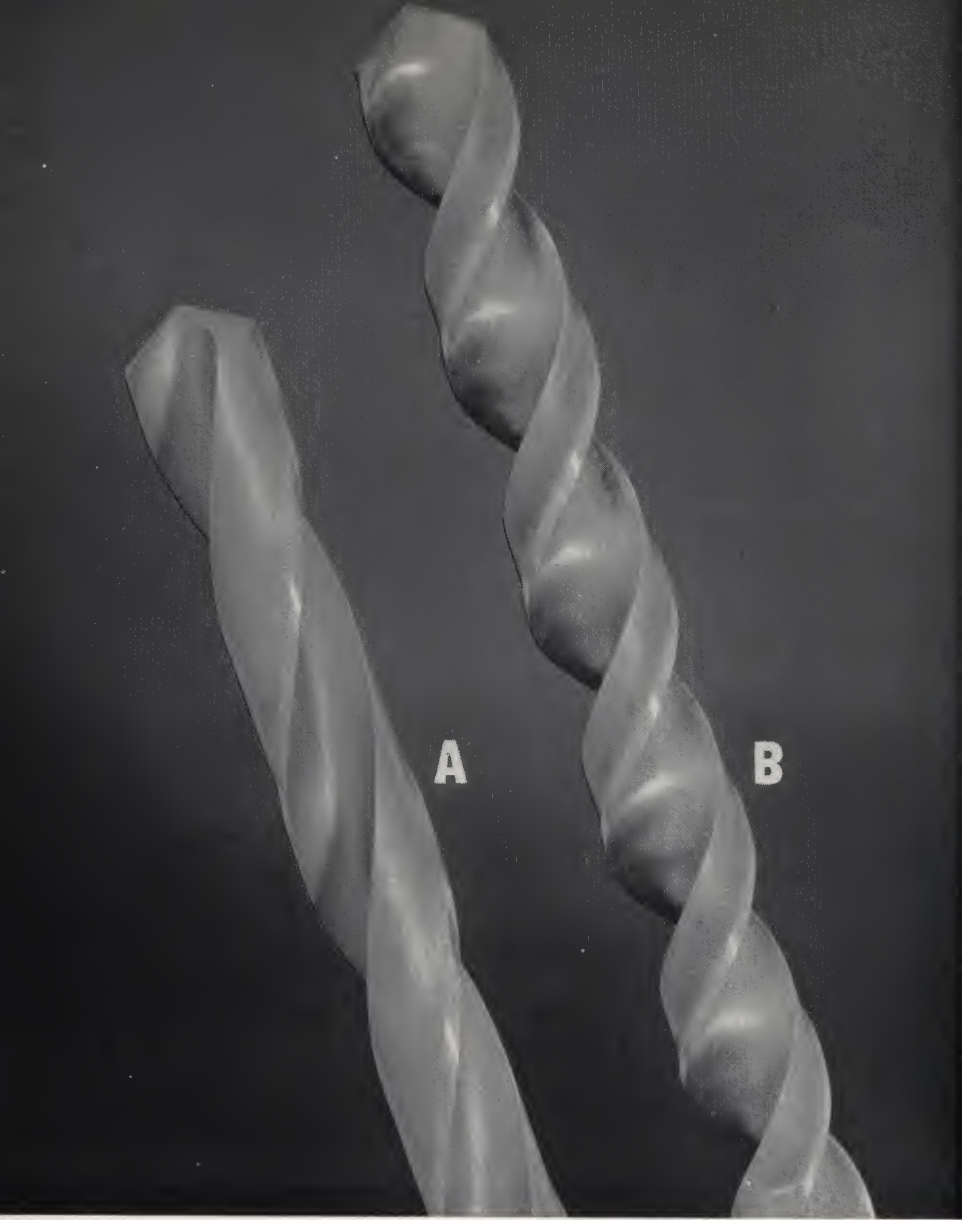


FIG. 10 — TWIST DRILLS: (A) *Double-fluted or standard twist drill, 24° spiral angle;* (B) *special double-fluted twist drill, 47° spiral angle.*

Twist drills similar to the standard drill but made with large, deeply cut flutes, with a polished finish, produce excellent results in machining all aluminum alloys. Such drills are used to a considerable extent for deep drilling, and, in the larger sizes, are provided with holes through the length of the drill to permit forcing cutting compound to the tip of the drill. These drills do not appear to be as strong as the ordinary twist drill, but less breakage is encountered with them because they cut so freely and the cuttings pass through the flutes so readily.

Speeds for drilling aluminum may range up to 600 peripheral feet per minute. The use of a large number of revolutions per minute for twist drills cannot be over-emphasized because the actual cutting speeds in feet per minute are necessarily low on most drilling equipment with the smaller sizes of drills. For instance, a ⅛-inch drill operating at 6,000 rpm has a cutting speed of only 200 feet per minute.

For hand feeding, a light feed is helpful, especially for small drills. For power feeds of various types of twist drills of high-speed steel, the feeds may be increased with the diameter of the drill. A feed of 0.004 to 0.012 inch per revolution may be used for drills up to ⅜-inch diameter; 0.006 to 0.020 inch per revolution for drills ⅜ to 1¼-inch diameter; and 0.016 to 0.035 inch per revolution for drills over 1¼-inch diameter.

When the work revolves and the drill is stationary, as in a lathe, the straight-fluted drill will sometimes give better results than the spiral-fluted drill.

The usual angle of about 59 degrees for the drill point, as supplied by drill manufacturers, is satisfactory for most jobs. The lip clearance of 12 to 15 degrees may be increased, particularly when the feed is heavy or when drilling the softer alloys. It may be advantageous to thin the point of the drill in order to reduce the pressure required to feed it, and to prevent overheating and oversize drilling.

Thin material is often drilled satisfactorily without the use of cutting compound. A copious quantity of cutting compound, however, should be applied to the drill when drilling deep holes, and it may be necessary to withdraw the drill from the hole occasionally

FIG. 11—REAMERS: (A) *Plain straight-fluted reamer;* (B) *spiral-fluted expansion reamer;* (C) *straight-fluted taper reamer;* (D) *spiral-fluted taper reamer.*

to apply cutting compound to the drill point and to dispose of the cuttings.

If drills break frequently, the trouble may be caused by lack of rigidity in the machine or work, an excessive feed, or insufficient lip clearance.

REAMERS

Most of the different types of reamers may be used for aluminum. The flutes may be straight, but spiral flutes frequently produce better results. (See Figure 11.) Flutes spiralled in the direction of rotation of the tool cut freely but feed into the work too rapidly. Therefore, reamers with the spiral opposite to the direction of rotation are generally preferred; this type cuts more slowly but the operation can be controlled better. In some instances, special reamers made with the alternate teeth spiralled in opposite directions have been found advantageous. The cutting edges of reamers should be finished by honing.

Machine reamers less than 2 inches in diameter may be operated at cutting speeds up to 400 feet per minute for reaming straight holes. For tapered holes, speeds up to 300 feet per minute may be used. Holes that are to be reamed should always be slightly undersize so that the reamer has a definite cutting action. If the hole is too near the finished size, an undesirable burnishing action may result.

SAWS

It should be emphasized that the same principles which govern the shape of cutting tools for aluminum should be applied, as far as practical, to saws for aluminum. While this statement may seem obvious, experience with sawing equipment indicates that these principles are often overlooked. It is especially important to use comparatively coarse teeth with curved gullets, which are free from sharp corners, burred edges, and rough surfaces to which cuttings may adhere. Clearance, although necessarily small, must nevertheless be provided; otherwise, the sides of the teeth will drag and

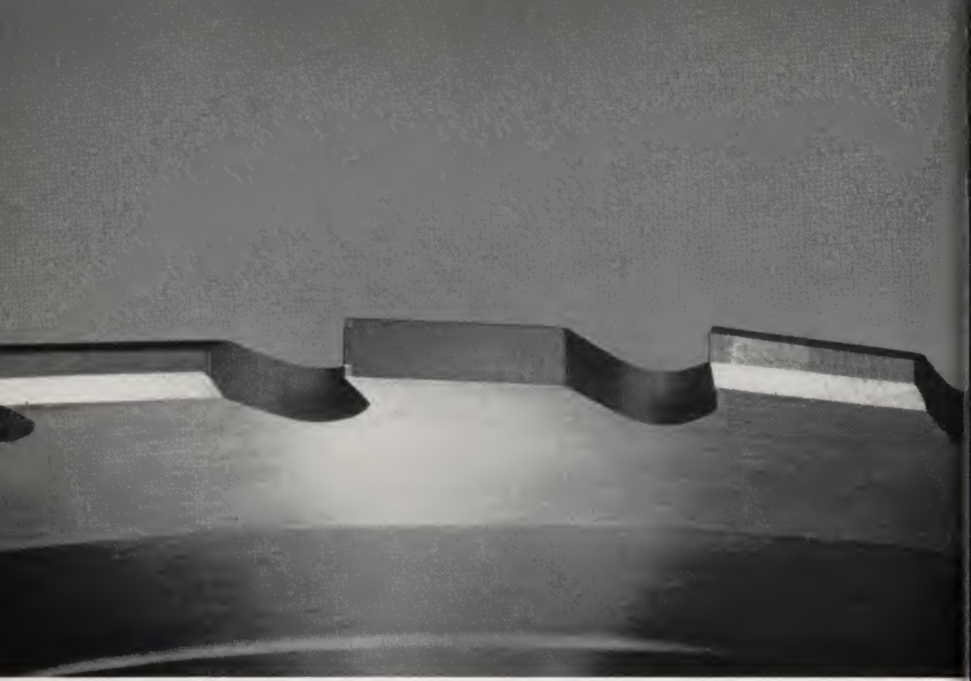
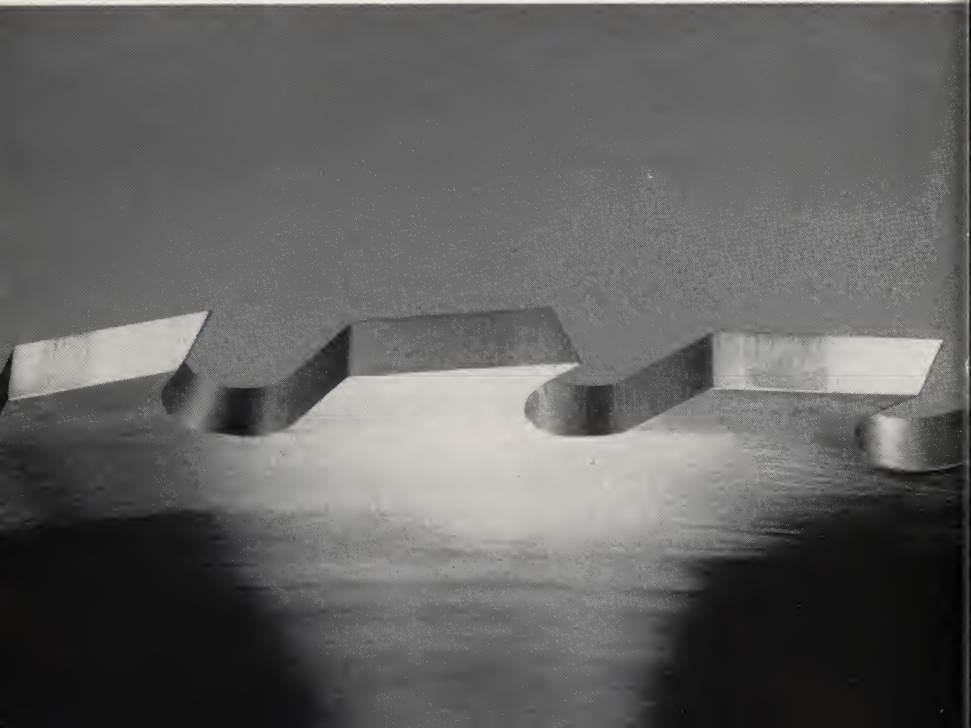


Figure 12 — *Chip-breaker type saw teeth.*

Figure 13 — *Alternate side rake type saw teeth.*



generate heat. These principles can, in general, be followed in circular saws but their application may be limited in the selection of band saws.

Circular saws with broad teeth are extensively used for aluminum. The teeth are broadened by swaging so that they cut wide enough to provide clearance for the saw blade in the saw cut. This type of tooth, however, does not provide satisfactory clearance for the tip of the tooth itself. Hence the sides of the teeth rub in the saw cut and generate heat. This type of saw blade is used largely for cut-off operations where the cuts are short and intermittent; for longer cuts it may be necessary to apply considerable cutting compound to keep the blade from overheating.

Circular saws with chip breaker teeth are better than the broad tooth type. This type of saw, illustrated in Figure 12, has teeth so profiled that one cuts deep and the next one cuts wide. Another preferred type of saw blade has alternate side rake teeth. The teeth are alternately arranged so that one cuts on one side and the next one on the opposite side of the saw cut. (See Figure 13.) These teeth should have a side rake of about 15 degrees.

For these types of blades, the top rake, or "hook" may vary up to about 45 degrees. When employing such a large rake angle, a tooth with a rather small included cutting angle results. While such teeth do not appear to be very strong, experience indicates that, with other conditions correct, the teeth will perform longer without resharpening than when a small rake angle is employed. Obviously, with the use of a large rake angle, the sawing machine must be sturdy and free from vibration, and the work must be securely clamped and fed with a positive feeding device. Where it is desired to feed the work by hand, the saw teeth should have little, if any, top rake; otherwise, the saw will enter the work too rapidly. Saw teeth for aluminum, however should never have a negative rake.

Circular saws are generally made of semi-high-speed steel or high-speed steel, or with teeth tipped with cemented carbide. They may be operated at peripheral speeds of 10,000 to 15,000 feet per minute. Speeds in the higher range apply to the carbide tipped saws. These saws produce excellent results on aluminum and they are finding

wide use, especially in large circular saws used for cutting heavy sections.

Band saws of the high-speed, metal-cutting type perform satisfactorily on aluminum. For light work, however, heavy-duty woodworking band saws may be used. Cutting speeds range from 2,000 to 5,000 feet per minute, but higher speeds have been satisfactorily attained. When using high sawing speeds it is, of course, important to have a machine that is free from vibration in order to minimize fatigue failure of the blade or teeth.

Generally, for light work, band saw blades made of spring tempered steel are used because the blades can be resharpened. For heavy work, the flexible back type of saw blade with hard teeth is preferred. Band saws should have relatively coarse teeth, about 8 teeth per inch, although this may vary, depending on the sawing conditions. A comparatively fine feed should generally be used to avoid large cuttings. The feed, however, will depend on the kind of work being done, especially the size, thickness or shape of the material sawed.

For circular or band saws, a cutting lubricant or coolant is necessary for most operations, especially for heavy sections. Soluble oil cutting compounds have been found satisfactory and ordinary mineral base lubricating oils may also be used, fed to the sides of the saw blade. An occasional application of paraffin wax or heavy grease will provide ample lubrication for some work. The life of band saws, in some instances, has been prolonged considerably by providing a slotted block through which the saw blade passes. This is arranged with a screw-operated grease gun to feed ordinary cup grease into the block so that the blade is continually passing through the supply of grease, which is supplied by an occasional turn of the gun's feed-screw.

Hack saw blades of the wavy-set type are well suited for cutting aluminum by hand.

ABRASIVE CUT-OFF WHEELS

The use of rubber-bonded abrasive cut-off wheels is a recent development which has been suggested for aluminum casting

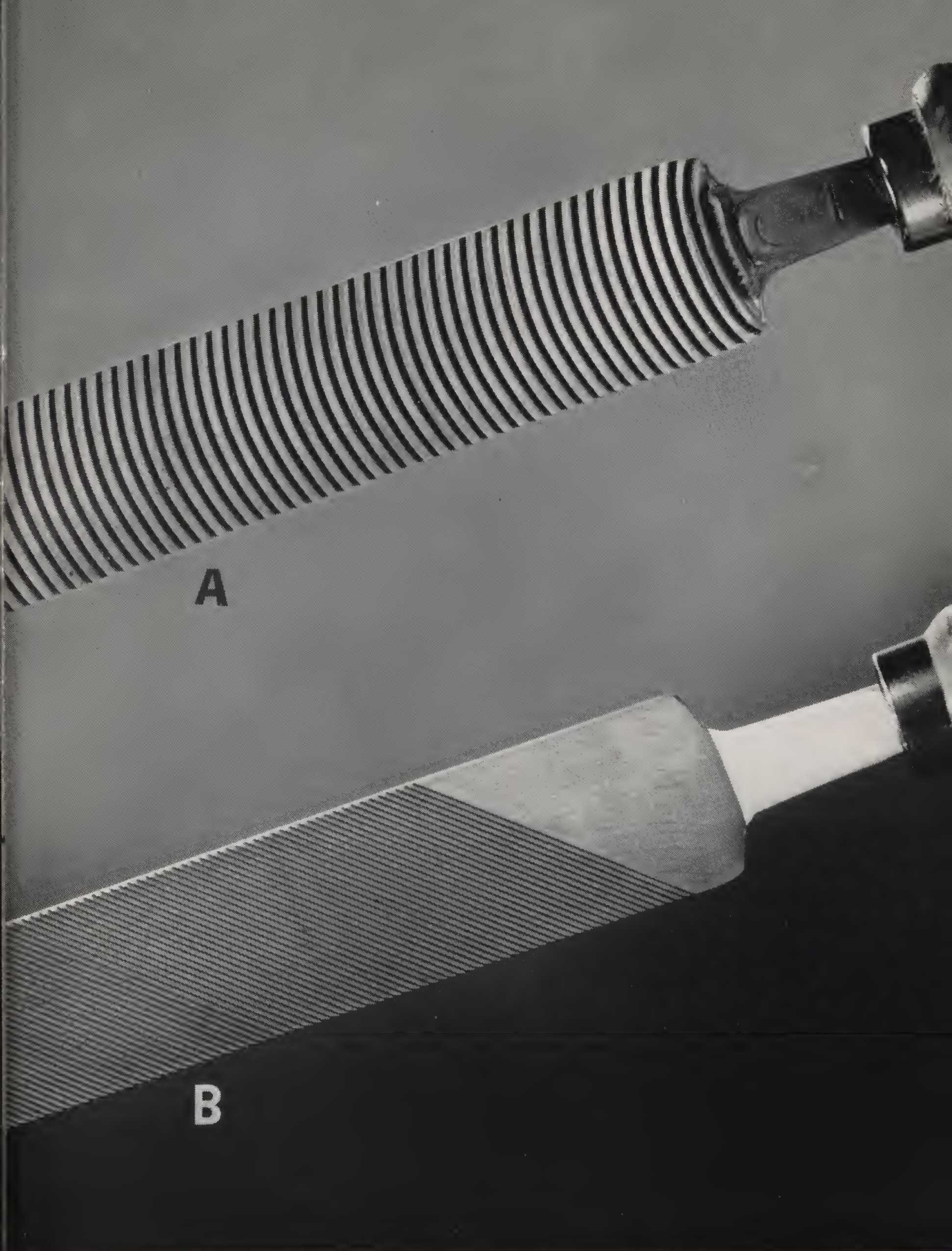


FIG. 14 — FILES: (A) *Deeply cut, coarse tooth file;* (B) *long-angle lathe file.*

alloys, and is already being used successfully for cutting heavy forging stock of the high silicon alloy 32S. The wheels tend to load when used to cut the softer wrought alloys such as A51S. Smooth, accurate cuts can be produced quickly and cheaply with these rubber-bonded wheels. They operate at speeds of 9,000 to 13,000 surface feet per minute and should be used with a voluminous supply of cutting fluid, preferably a water soluble oil at a dilution of about 1:80.

FILES

An excellent general purpose file for aluminum is shown in Figure 14A. It has coarse, deeply cut, curved teeth. Its pitch of about 10 teeth per inch affords good clearance for the filings; hence, no difficulty is experienced with loading of the teeth. This file removes metal rapidly and also produces a smooth surface. Because of the coarse teeth, it cannot be used for filing small surfaces. Files similar to "A," but with the teeth notched to break up the cuttings, remove metal even more rapidly, but they do not produce quite so smooth a finish. Certain other modifications of this file, with double-cut teeth (more like those of a rasp), have been found unsatisfactory, in that they cut poorly and produce a rough surface.

The long-angle lathe file, illustrated in Figure 14B, is excellent for finish-filing aluminum. It has finer teeth than those of file "A," which reduce the tendency to "run off." Files of this type may have a pitch of 14 to 20 teeth per inch, and the teeth should be cut to a side rake angle of about 45 to 55 degrees. The side rake angle provides a more efficient cutting action by producing a slicing motion. Because of the large angle of the teeth, the direction of motion of the file is effective in driving the cuttings from the teeth.

Files with single-cut, fine teeth do not work well on aluminum, because the cuttings stick in the teeth. Those with double-cut teeth, whether fine or coarse, are no better on soft materials but perform fairly well on the harder materials. Ordinary files with single-cut, coarse teeth, however, may be made to perform nearly as well as the long-angle lathe file, by employing a side sweep motion instead of one in the direction of the length of the file. Files with coarse

or medium-coarse teeth, single cut, are better if used with oil. Chalk rubbed over the teeth also helps to prevent loading.

Where rifflers are used, they should have coarse teeth. Rotary files and burrs should be spiral-cut. Sanding drums of different sizes with sleeves of various grits are often used for special work.

CUTTING SPEEDS AND FEEDS

Wide ranges of cutting speeds and feeds may be used in machining aluminum and its alloys. The particular values for speed and feed are usually dependent on the character of the work, the type of tool, the lubricant, and the machine on which the work is done. The rate at which metal can be removed is limited mostly by the machine tool and aluminum can usually be machined to best advantage by using the highest speed at which the equipment is capable of operating, with moderate feeds and cuts. General information on speeds, feeds, and cuts applicable to average shop conditions will be found in Table 2, page 42, where a maximum cutting speed of about 1,000 peripheral feet per minute is indicated. On page 21, however, cutting speeds up to 15,000 feet per minute are suggested for sawing, and according to recent experiences, cutting speeds in the neighborhood of 8,000 feet per minute are common on machine tools that are designed especially for such operations as milling and routing.

For high speeds, the machine tool must have a heavy, sturdy construction and be balanced to operate smoothly at the speeds involved. It is also important to have much more power than is available in the average machine tool. For face milling cutters, 4 to 6 inches in diameter, it is not uncommon to use a 20 horsepower motor. Under some conditions, a 40 horsepower motor may be required to drive the cutter.

The feed and depth of cut must be so related to the cutter speed and number of teeth that each blade takes a definite bite. Each blade should advance into the work at least 0.003 inch. If the cut per blade is too light, the effect may be to burnish the work and generate heat. In some instances, an advance per cutter blade up to about 0.015 inch has been used. The depth and width of cut may be large and the

feed may be many times faster than is possible in the ordinary milling machine. Feeds as high as 20 feet (240 inches) per minute have been achieved.

Milling cutters should have fewer cutter blades or teeth than is customary for ordinary milling practice. In some instances, two blades have been found satisfactory for a 6-inch diameter cutter. The rake angles of the cutting tools will correspond approximately to those indicated for general machine shop practice with a tendency towards the small end of the angle range in order to give sturdy teeth. Clearances may be slightly less, or about 5 to 7 degrees. The cutting edges should be keen and smoothly finished. Extra time in polishing the surfaces is well spent. High-carbon steel, high-speed tool steel, and the cemented-carbide type tools have all been used successfully. In some instances, the plain high-carbon steel tools, when properly finished, have been found superior to the other types.

A coolant is generally necessary; soluble oil is most used, mixed with a larger proportion of water than is customary for ordinary machining practice. The liquid must be supplied in a large volume, so that the work is practically submerged. Sometimes the coolant is introduced with a stream of compressed air. On the other hand, some operations have been accomplished successfully without a coolant, using a stream of compressed air to cool and blow away the chips. Indications are that with proper finish of the tool, suitable rake angles, and the correct amount of cut taken with each blade, the chip is parted from the metal so rapidly that most of the heat generated is concentrated in the chip, which is thrown from the machine fast enough to prevent any serious heating of the tool or the work. Removal of the large volume of hot chips is a problem that may require special handling methods.

CUTTING COMPOUNDS

The free-cutting aluminum alloys (Type I, Table 1) are frequently machined without a cutting compound, especially for roughing operations. When heavy cuts and feeds produce excessive heat, however, a cutting compound should be used, and often the

type of compound that is essentially a coolant will be satisfactory. For this purpose, soda water or soluble oil is generally employed, and, in some instances, it may be desirable to add a small amount of lard oil or kerosene. This type of compound is widely used for milling, drilling, and sawing operations.

Where the cutting compound must have more definite lubricating characteristics, the following are suggested:

1. There are a number of ready-mixed cutting lubricants intended for use as supplied by the producer. Most of these, however, are generally blended by the user with a low viscosity mineral oil and the blend may be varied to suit different machining operations.
2. A satisfactory cutting lubricant may be prepared using mineral oil with the addition of 5 to 10 per cent fatty oil such as lard oil. This is widely used for automatic screw machine work.
3. Another excellent lubricant consists of equal parts of kerosene and lard oil, but the proportions may be varied over a wide range for different operations.

With heavy cuts and slow feeds, such as for roughing or tapping operations, the cutting lubricant should be of high viscosity.

GRINDING

The free-cutting aluminum alloys may be ground, polished, or buffed readily. Experience with aluminum indicates that good results are obtained with commercial silicon-carbide grinding wheels such as Crystolon, Carborundum, and Natalon. However, the advice of the supplier should be obtained in selecting the proper grade of each commercial make of wheel.

Once a grinding wheel has been selected, there are three variables which affect the quality of a finish, namely, wheel speed, work speed, and grinding compound. Wheel speeds of about 6,000 feet per minute have given good results, but both wheel and work speeds can best be set by the experienced operator according to his own good judgment. A solution of soluble cutting oil and water works well as a grinding compound. It is important that the fine

grindings of aluminum be strained from the compound before re-using, in order to prevent deep scratches on the finished surface.

The soft alloys cause the grinding wheel to clog and require generous use of a grease stick. Furthermore, special care may be required in grinding castings and wrought alloy products that have been heat treated. Their greater resistance to cutting generates a considerable amount of heat which, in turn, may cause warping and render the maintenance of dimensions difficult.

Good grinding results are usually obtained when the procedures indicated in Table 3, page 43, are employed.

FINISHES

Smooth, finished surfaces may be produced on aluminum and most of its alloys by machining. Brightly polished surfaces may be produced by buffing. Other effective mechanical finishes include sandblasting, scratchbrushing, and hammering. Aluminum may also be finished by the use of chemical solutions and by electrochemical processes. Alrok* is a well-known protective finish of the former type and Alumilite* is a popular electrochemical or anodic finish. With the latter type of finish, aluminum may be attractively colored.

Paint, lacquer and enamel also are applicable to aluminum. For more complete information on the subject of finishing see the booklet, "Finishes for Aluminum," available on request.

SOME SPECIAL CHARACTERISTICS OF ALUMINUM

1. Weight—Aluminum weighs only about one-third as much as steel; hence, rather large castings and forgings can be handled without the use of special handling equipment. Owing to its light weight, inertia forces are less in machining operations in which the work moves.

2. Thermal Expansion of Aluminum—Aluminum alloys have higher coefficients of expansion than many other commercial

*Patented Process.

metals. Therefore, warping and distortion may occur if a cast or forged part is excessively overheated during machining. For precise work, when appreciable heating does occur, the work should be cooled before it is finished to size and calipered so that the required final dimensions may be obtained.

Overheating may arise from the use of improperly designed or dull tools, from failure to use a lubricant where indicated, or from the use of heavy feeds or cuts.

Distortion also is sometimes caused by improperly clamping the work in the machine.

3. Finish Allowance—If sand castings are to be machined, allow sufficient stock for finishing. Little or no machine finishing is required for permanent mold castings, forgings, or other wrought products.

4. Value of Cuttings—Because of the value of aluminum cuttings and scrap material, they should be protected from contamination. When a premium is paid for controlled purity or composition of scrap, the cuttings from different alloys should be segregated.

5. Chip Clearance—Provide ample room for chip clearance.

6. Holding Work Rigidly—In utilizing the high speeds recommended for machining aluminum, the work must be held rigidly and maintained free from vibration.

7. Foreign Inclusions—The presence of foreign inclusions in castings may seriously reduce tool life.

2

Automatic Screw Machine Practice

TOOL MATERIALS

THE same tool materials that are used for general machining, as indicated in Part I, are used for automatic screw machine tools. Some form of high-speed steel is most generally used. However, plain high-carbon steel may be more economical for fragile tools, such as small drills and taps, and for tools operated at low cutting speeds. Cemented carbide tools have proved economical for many screw machine operations, particularly where high speeds prevail. Substantial increase in tool life between grinds is usually obtained. However, operating conditions must be suitable for such tools, as lack of rigidity and interrupted cuts may cause unsatisfactory results.

STOCK FOR AUTOMATIC SCREW MACHINE WORK

The four alloys of Alcoa Aluminum most commonly used for screw machine stock are 11S-T3, 17S-T, 24S-T, and 53S-T. Alcoa 11S-T3 is a free machining alloy developed especially for use in automatic screw machines; it produces an excellent finish when machined with tools having little or no top rake, and the chips are fine and broken. Alcoa 17S-T and 24S-T alloys have properties which recommend them as general purpose materials and are widely used. The disposal of the relatively long chips from these alloys is sometimes a problem, but by the proper combination of feeds, speeds, and tooling, the chips may be materially reduced in size. Alcoa 53S-T alloy is specified when the product is subject to conditions requiring the superior corrosion resistance of this alloy. The chips from this alloy are usually long and stringy, and more difficult to break up.

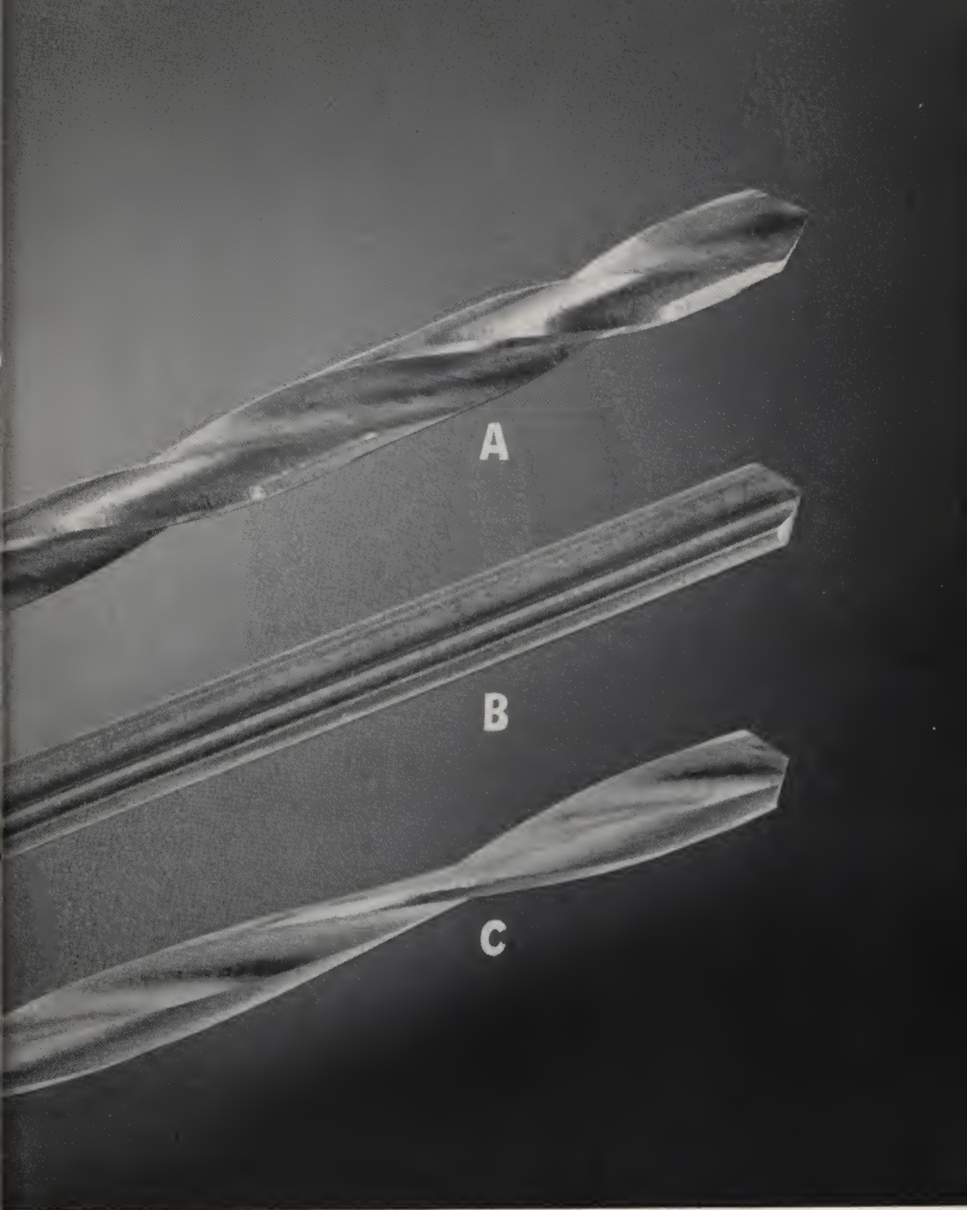


FIG. 15 — AUTOMATIC SCREW MACHINE DRILLS: (A) *Standard drill;* (B) *straight-fluted drill;* (C) *drill with large chip clearance.*

DRILLS

Standard twist drills are generally satisfactory for drilling holes to a depth of 5 to 6 diameters in aluminum alloys. For drilling deeper holes, special drills, such as straight-fluted drills or those which are used to drill phenolic resins, may be more satisfactory. These drills are shown in Figure 15. High-spiral drills with a spiral angle of 47 degrees are often used in drill presses or for other secondary operations on deep holes. (Page 16.) All drills used for aluminum should have narrow lands and large, smooth, highly polished flutes to allow the chips to pass out readily. Drills are generally ground with an included lip angle of 118 degrees and a clearance angle of 12 to 20 degrees. In some cases, it is desirable to use a greater included angle to produce a narrow chip which will pass up the flutes more readily. For straight-fluted drills, the clearance angle should be 20 degrees or slightly more. As in the case of other metals, it is often desirable to thin the drill web, particularly as the drill is ground back.

In drilling very deep holes, it is necessary either to use more than one drill, or to pull out the drill a sufficient number of times. The drill should be entirely withdrawn from the hole to permit removal of the chips and to allow the cutting compound to flow on the drill point. Safe working limits for drill depth are as follows:

First entrance:	4 diameters
Second entrance:	2 diameters
Third entrance:	1 diameter
Subsequent entrances:	½ to 1 diameter

In some cases, these limits may be exceeded, depending on the type of drill, its size, the feed, and the ease with which the cutting compound can enter the hole. As the cutting compound cannot penetrate readily to the bottom of a deep hole, the feed of the drill should be somewhat less than for shallow depths.

FORM AND CUT-OFF TOOLS

For machining 11S-T3, a top rake of from 0 to 3 degrees usually produces the best results and breaks up the chips satisfactorily. For

machining 17S-T and 24S-T, a top rake of from 5 to 15 degrees is generally necessary to produce a free cutting tool. For the softer alloys, a greater rake angle is desirable. This is sometimes obtainable by grinding a groove in the tool parallel to the cutting edge. Form tools are usually made with a one-half degree side clearance to aid in securing a smooth finish on the side of flanges and grooves. In screw machine practice, front clearance angles are determined by the design of the machine and tool holder, rather than the material being machined. The angles used in common screw machine practice are satisfactory for all aluminum alloys.

Figures 16 and 17 show the relation of the tools to the work.

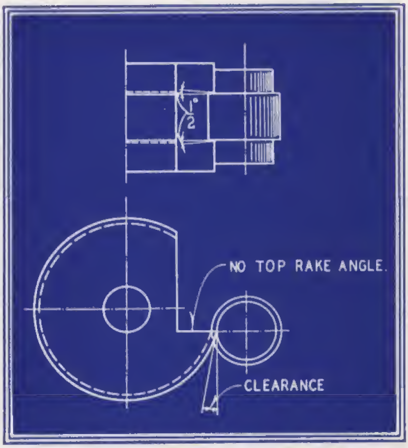


Figure 16 — *Circular form tool for 11S-T3 alloy.*

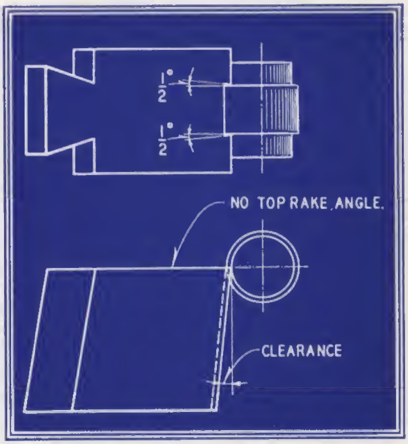


Figure 17 — *Dovetail form tool for 11S-T3 alloy.*

It is frequently possible to break up chips from aluminum alloys with various types of chip breakers. If this practice is objectionable, they may sometimes be broken up by notching the cam. In such cases, the cam should be designed so that, after the tool drops back into the notch, it starts forward again at the same cutting feed before actual cutting is resumed. If this is not done, the tool will tend to dive into the work.

Every material has a limit beyond which it is not practical to form with a side form tool without support, because the side pressure becomes so

great that the part being formed will spring or twist off. This limit is usually expressed as a ratio of the form length to the smallest form diameter. This ratio is 2¼ for 11S-T3 and 53S-T, and 2½ for 17S-T and 24S-T.

A front angle of 22 degrees on a cutoff tool will generally reduce the cutoff burr to a minimum. In some cases where springing of a

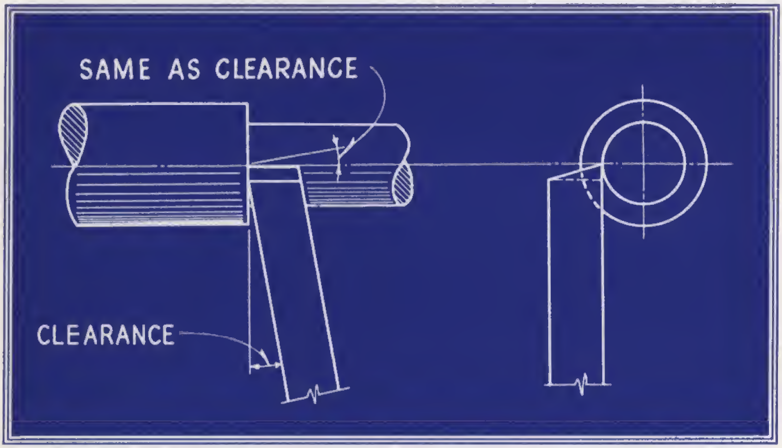


Figure 18 — *Box tool for machining Alcoa 11S-T3 alloy.*

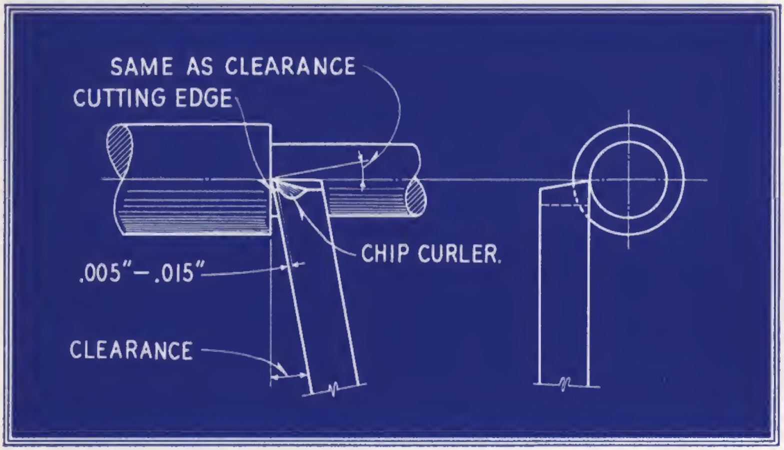


Figure 19 — *Box tool for machining Alcoa 17S-T and 24S-T alloys.*

straight cutoff tool is encountered, this spring may be minimized by reducing the front angle to approximately 15 degrees.

BOX TOOLS

When machining aluminum with box tools, even when taking a heavy cut, smooth finishes and accurate tolerances are obtained—roughing cuts, then, are necessary only when a large amount of metal must be removed. In those cases where a roughing cut is necessary, a substantial amount is left for the finishing cut.

Box tool holders are designed so that the tool is usually set with a clearance angle of 8 degrees. This means that the section of the tool bearing on the turned diameter should be ground on an angle equal to this clearance angle as shown in Figure 18. This results in an included cutting angle of 82 degrees. When the tool is set, the bottom edge of the tool should be parallel to the axis of the work. The cutting edge may be ground square with the tool shank, or with a slight angle, depending upon whether a square shoulder is necessary on the work. Such a tool produces a smooth finish and well broken up chips when machining 11S-T3 alloy.

Figure 19 shows a box tool ground for cutting 17S-T and 24S-T alloy. The addition of a hook or groove is the only variation from the tool used for 11S-T3 alloy. This groove provides more rake to the cutting edge, producing a freer cutting tool. The groove also forms the chips into a helical coil and guides them out of the holder and away from the work. If a long coiled chip is objectionable, the groove may be converted into a chip breaker by reducing its width and depth as shown in Figure 20.

THREADING TOOLS

Taps having a smooth finish are highly desirable in machining aluminum. Therefore, the ground thread type is preferred. Straight-fluted taps are most generally used in automatic screw machines. When chips remaining in the tapped hole are a problem, it may be advisable to use spiral-fluted taps. All flutes should be large and smoothly finished to allow the chips to pass out readily. Flutes

should be shaped so as to provide a positive rake angle at the cutting edge. Lands should be relatively narrow to hold friction to a minimum. No eccentric relief is necessary if width of lands is not excessive.

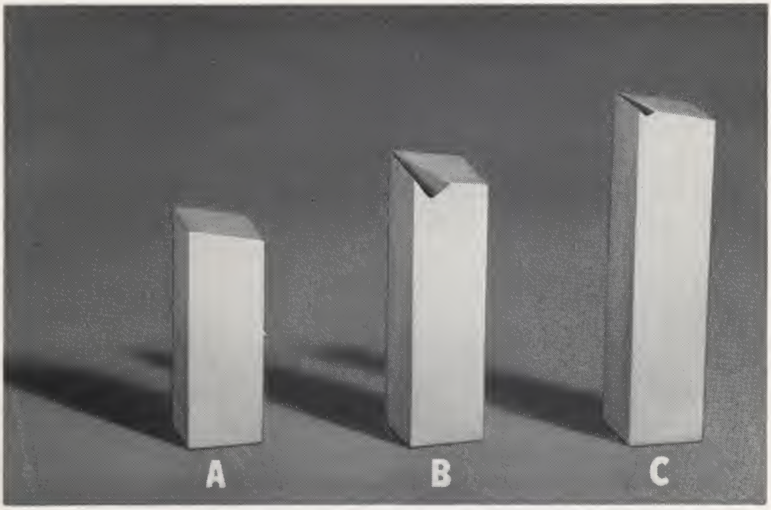


Figure 20 — *Three types of box tool bits:* (A) *for machining Alcoa 11S-T3 alloy;* (B) *for Alcoa 17S-T and 24S-T alloys;* (C) *tool with chip breaker groove.*

Taps having two flutes are usually preferable up to ⅜-inch diameter. For larger sizes, the number of flutes, as recommended by the manufacturer, has been found generally satisfactory. In most cases, a chamfer angle of approximately 20 degrees gives good results. For tapping to within 1½ threads of a shoulder, a chamfer angle of 35 degrees is required. The chamfer clearance angle should be 4 to 5 degrees. For tapping the softer alloys, increased rake angle may be necessary at the cutting edge. In all cases, care should be taken to remove all grinding burrs and feathery edges.

Chasers for machining aluminum should also have an excellent finish. For 11S-T3, 17S-T, and 24S-T a straight hook of from 10 to 15 degrees will usually produce good threads. (See Figure 21.) For 53S-T, a greater angle may be advantageous. A chamfer angle of

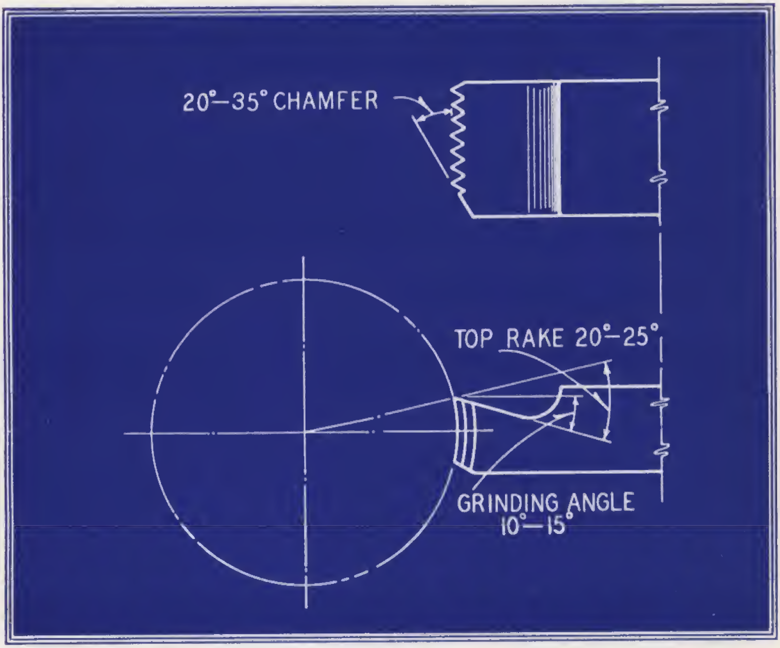


Figure 21 — *Chasers for self-opening die heads.*

approximately 20 degrees is generally used. When threading to within 1½ threads of a shoulder, it is necessary to use a chamfer angle of 35 degrees. A clearance angle on the chamfer of from 6 to 7 degrees is desirable. A spiral angle ground into the front of the chasers, similar to the top rake on spiral-pointed taps, as shown in Figure 9, will force chips ahead of the tool and minimize loading of the die head with chips. In all cases, it is desirable to remove grinding burrs and feathery edges from the chasers to produce best results.

When ordering taps or chasers, it is advisable to specify that they are to be used on aluminum.

CHIP BREAKERS

Chip breakers are sometimes ground into tools for machining screw machine alloys, other than 11S-T3. Two main types of chip

breakers are used, the step and the groove types. The step type is produced by grinding a step in the tool, so that the chip slides over the cutting edge and hits against the shoulder. This tends to coil and break the chip. The groove type consists of a groove ground slightly back of the cutting edge which also tends to coil and break the chip. The exact shape of the groove is not critical, but the chip breaker should be shallow and narrow to bend the chip before it would normally tend to coil. If this is not done, the chip will not break and long coils will result. Figures 22 and 23 illustrate these types of chip breakers as applied to form tools.

SURFACE SPEED OF STOCK

The lighter weight of aluminum, about one-third that of other metals, is an advantage when using high spindle speeds. Its lower inertia causes less wear on the spindles, belts, and motors, especially when quick changes in speed are involved.

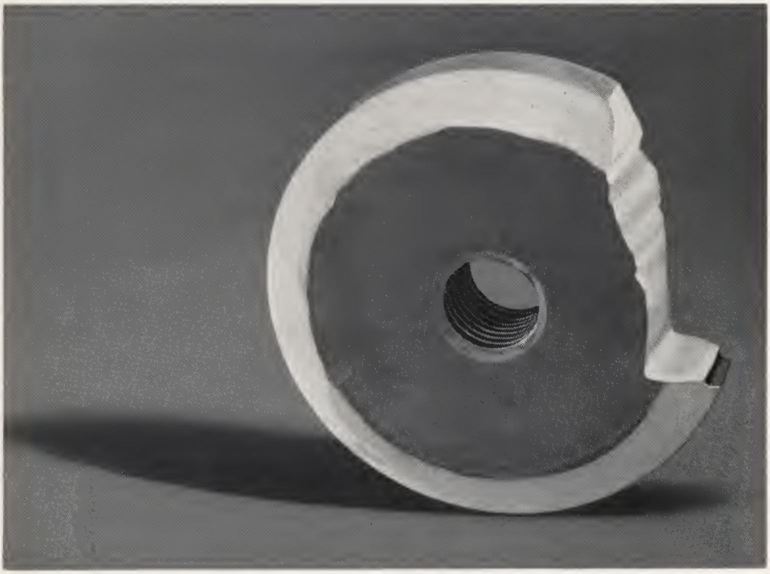


Figure 22 — *Circular form tool with step-type chip breaker.*

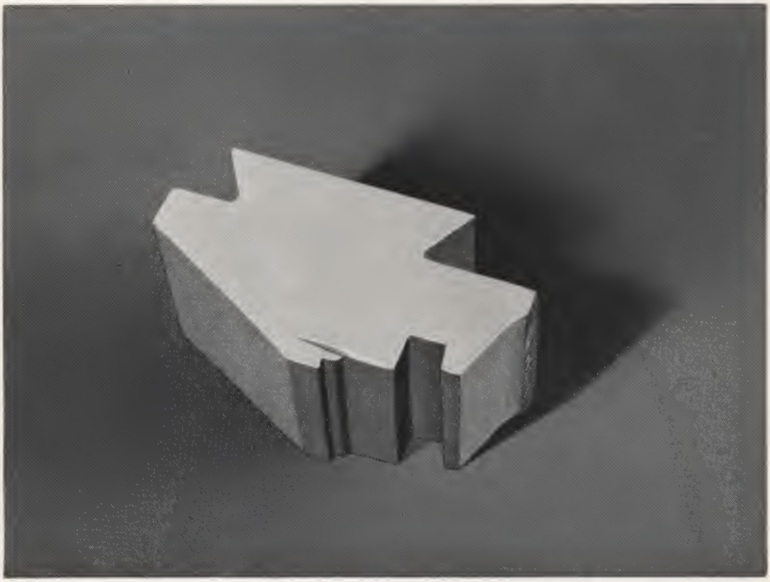


Figure 23 — *Form tool with groove-type chip breaker.*

Aluminum alloys can usually be machined using the maximum spindle speed available in all standard types of automatic screw machines for turning, drilling, forming and cutting off. They have been machined in excess of 1,000 stock surface speed per minute, when using cemented carbide tipped tools with no indication that such speed is excessive. How much this can be exceeded has not been determined.

For threading and tapping operations, each type of machine has a suitable speed, which is considered good practice for machining aluminum, as well as other metals. This speed is usually not more than one-third of that used for other operations.

TOOL FEEDS

Tool feeds vary with cutting conditions as well as with requirements of tolerance and finish. Overlapping operations and the condition of the machine may require lower feeds than normal. The

feeds shown in Table 4, therefore, are only approximate and must be altered to meet each situation. Generally, feeds approximately 15 per cent higher than those given in Table 4 for aluminum alloys may be used for 11S-T3.

CUTTING COMPOUNDS

Generally, cutting compounds which are used for other metals are suitable for Alcoa Aluminum alloys. There are a number of good commercial cutting oils prepared especially for machining aluminum which gives satisfactory results. Usually a light mineral oil with about five per cent of fatty additions with a viscosity of 56 Saybolt seconds at 100°F. is used. It is desirable to have a flash point of over 260°F. As with all metals, it is essential that an ample supply of cutting oil be provided. A large volume flowing over the tools is more desirable than a high pressure stream.

TABLE 1—COMMERCIAL ALUMINUM ALLOYS

Type*	Alcoa Alloy	COMPOSITION (per cent)				
		Cu	Fe or Mn	Si	Mg	Other
		Non-Heat-Treated Casting Alloys				
I	173	7.0	......	...	...	2.0 Sn
	C113	7.5	1.2 Fe	4.0	...	2.0 Zn
	645	2.5	1.5 Fe	...	...	11.0 Zn
	B113	7.5	1.2 Fe	1.5	...	......
II	112	7.5	1.2 Fe	...	...	2.0 Zn
	216	....	......	...	6.0	......
	A214	....	......	...	3.8	2.0 Zn
	109	12.0	......	...	...	......
	12	8.0	......	...	...	......
	214	....	......	...	3.8	......
	212	8.0	1.0 Fe	1.2	...	......
	B214	....	......	1.8	3.8	......
III	172	7.8	......	2.5	...	......
	A108	4.5	......	5.5	...	......
	108	4.0	......	3.0	...	......
	356	....	......	7.0	0.3	......
	43	....	......	5.0	...	......
		Heat-Treated Casting Alloys (a)				
I	220(b)	....	......	...	10.0	......
	122	10.0	1.2 Fe	...	0.2	......
II	D195	5.5	......	0.7	...	......
	142	4.0	......	...	1.5	2.0 Ni
	195	4.0	......	...	...	......
	B195	4.5	......	3.0	...	......
III	355	1.3	......	5.0	0.5	......
	A355	1.4	0.8 Mn	5.0	0.5	0.8 Ni
	356	...	......	7.0	0.3	......
(c)	A132	0.8	0.8 Fe	12.0	1.0	2.5 Ni
		Heat-Treated Wrought Alloys (a)				
I	11S(d)	5.5	......	...	...	0.5 Pb+0.5 Bi
II	17S(d)	4.0	0.5 Mn	...	0.5	......
	24S(d)	4.4	0.5 Mn	...	1.5	......
	25S	4.5	0.8 Mn	0.8	...	......
	70S	1.0	0.7 Mn	...	0.4	10.0 Zn
	18S	4.0	......	...	0.5	2.0 Ni
	14S	4.4	0.8 Mn	0.8	0.4	......
III	61S	0.25	......	0.6	1.0	0.25 Cr
	53S(d)	...	......	0 7	1.3	0.25 Cr
	A51S	...	......	1.0	0.6	0.25 Cr
(c)	32S	0.8	......	12.0	1.0	0.8 Ni
		Non-Heat-Treated Wrought Alloys				
II	56S	...	0.1 Mn	...	5.2	0.1 Cr
III	52S	...	......	...	2.5	0.25 Cr
	3S	...	1.2 Mn	...	...	......

* Indicates relative machinability. Type I alloys have best machining characteristics.
(a) Heat treated as usually sold, namely a solution treatment followed by aging at room or elevated temperature.
(b) Alloy 220 is not aged.
(c) Alloy cuts freely, but wear on tools may be excessive unless they are tipped with cemented carbide.
(d) Used on automatic screw machines.

TABLE 2—CUTS, SPEEDS, AND FEEDS WHEN MACHINING ALUMINUM ALLOYS

(See Note)	ROUGH MACHINING			FINISH MACHINING		
	Max. Cut Inches	Speed (fpm)	Feed, Inches	Cut, Inches	Speed (fpm)	Feed, Inches
LATHE TURNING						
Type I castings, not heat treated	0.25(a)	500 to 900	0.020 to 0.030	0.002 to 0.010	Maximum	0.002 to 0.010
All others	0.19(a)	400 to 800	0.007 to 0.020	0.002 to 0.010	600 to 900	0.002 to 0.010
MILLING						
Type I castings, not heat treated	0.25	400 to 600(b) 500 to 700(c) Maximum (d)	5 to 15(e)	0.010 to 0.020	500 to 700(b) 500 to 700(c) Maximum (d)	10 to 25(e)
Type I castings, heat-treated Type II castings Types I and II wrought alloys, heat-treated	0.25	400 to 600(b) 500 to 700(c) Maximum (d)	4 to 10(e)	0.010 to 0.020	500 to 700(b) 500 to 700(c) Maximum (d)	5 to 15(e)
Type III alloys	0.25	300 to 500(b)	3 to 8(e)	0.010 to 0.020	500 to 700(b)	4 to 10(e)
BORING						
Light duty (1 to 2 inch)	0.09(a)	Maximum(f)	0.010 to 0.020	0.010 to 0.020(a)	Maximum(f)	0.001 to 0.005
Medium to heavy duty	0.25(a)	600 to 1000	0.007 to 0.015	0.010 to 0.020(a)	600 to 1000	0.001 to 0.003
SHAPING						
Heavy duty (36 inch)	0.25	Maximum(g)	0.010 to 0.030	0.005 to 0.010	Maximum(g)	0.100 to 0.150
PLANING	0.38	Maximum(h)	0.025 to 0.100	0.005 to 0.015	Maximum(h)	0.050 to 0.375

Note: See Table I for explanation of Type numbers listed above in first column.
(a) Cut measured on radius.
(b) For carbon steel tools.
(c) For high-speed steel tools.
(d) For cemented carbide tools.
(e) Travel of work.
(f) Peripheral speed of tool is maximum of most machines.
(g) Travel of ram.
(h) Speed of table.

TABLE 3—MECHANICAL FINISHING OF ALUMINUM ALLOYS

	ROUGHING			Greasing or Oiling	Buffing	Coloring	Finish* Grinding
	Solid Wheel	Cloth Belt	Sewed Muslin				
Abrasive	Al_2O_3 or SiC	Al_2O_3	Al_2O_3	Turkish Emery	Tripoli	Fine Lime or Soft Silica	SiC
Carrier	Solid Wheel	Cloth Belt	Sewed Muslin Buffs	Sewed Muslin Buffs	Pocketed Muslin Buffs	Open Muslin or Flannel	Solid Wheel
Grit	16 to 100	46 to 300	46 to 80	120 to 240			30 to 40
Bond	Phenolic Resin	Glue	Glue	Glue			Vitrified
Hardness	Medium		Medium	Medium	Soft	Very Soft	Soft
Peripheral Speed of Wheel, feet per minute	6,000 to 12,000	3000	6000	6000	7000 to 8000	3000 to 4000	6000 to 7000
Lubricant	Dry or Grease	Grease or Kerosene	Grease	Grease	Grease		Soluble Oil 1 to 35 or 40

* Mechanical finish applied to castings—to be preceded by machining.

TABLE 4—APPROXIMATE FEEDS FOR MACHINING ALCOA ALUMINUM ALLOYS

Tool	Cut Width or Depth Inch	Diameter of Hole Inch	Feed per Revolution Inch
BOX TOOLS	1/32		0.012
	1/16		0.010
	1/8		0.008
	3/16		0.008
	1/4		0.006
CENTER DRILLS		Under 1/8	0.004
		1/8 and over	0.008
CUT-OFF TOOLS			
Circular	3/64—1/16		0.003
	3/32—1/8		0.0035
	5/32—3/16		0.004
Straight	1/16—1/8		0.003
DRILLS		1/16	0.004
		3/32	0.006
		1/8	0.009
		3/16	0.011
		1/4	0.013
		5/16	0.014
		3/8	0.015
		1/2 and over	0.016
FORM TOOLS	1/8—1/4		0.002 —0.004
	3/8—1/2		0.0015—0.0035
	5/8—3/4		0.0015—0.003
	1		0.001 —0.0025
REAMERS	0.004—0.008	Under 1/8	0.007—0.010
	0.009—0.012	1/8 and over	0.010

TABLE 5—PHYSICAL PROPERTIES

Alloy	Specific Gravity	Weight, pounds per cubic inch	Electrical Conductivity, Per Cent of I. A. C. S.	Thermal Conductivity at 100° C. C.G.S. Units
17S-T	2.79	0.101	30	0.28
24S-T	2.77	0.100	30	0.28
11S-T3	2.82	0.102	40	0.37
53S-T	2.69	0.097	40	0.37

TABLE 6—TYPICAL MECHANICAL PROPERTIES[1]

Alloy	Yield Strength[2] (Set 0.2%) lb./sq. in.	Ultimate Strength lb./sq. in.	Elongation per cent in 2 inches	Brinell 500 Kg. 10 mm. ball	Shearing Strength[3] lb./sq. in.
17S-T	40,000	62,000	22	100	36,000
24S-T	45,000	68,000	22	105	41,000
11S-T3	42,000	49,000	14	95	30,000
53S-T	33,000	39,000	20	80	24,000

[1] Young's modulus of elasticity is approximately 10,300,000 pounds per square inch for aluminum alloys.

[2] Stress which produces a permanent set of 0.2 per cent of the initial gauge length. (American Society for Testing Materials Specification for Methods of Tension Testing, E 8-36.)

[3] Single-shear strength values obtained from double-shear tests.

TABLE 7—COMPARATIVE WEIGHTS OF STANDARD SIZES OF SCREW MACHINE STOCK[1]

11S-T3 and 17S-T ALLOYS[2]					BRASS AND STEEL (Round Wire and Rod)	
Diameter or Distance Across Flats—Inches		Standard Finish	Weight Lb. per Ft.[3]		Weight Lb. per Ft.	
Fraction	Decimal		Hexagonal	Round	Brass	Steel
1/8	.125	Drawn	......	0.0149	0.045	0.042
9/64	.141	Drawn	......	0.0190	.....	.....
5/32	.156	Drawn	......	0.0233	.....	.....
11/64	.172	Drawn	......	0.0283	.....	.....
3/16	.188	Drawn	0.0372	0.0336	0.102	0.094
13/64	.203	Drawn	......	0.0396	.....	.....
7/32	.219	Drawn	0.0506	0.0461	.....	.....
15/64	.234	Drawn	0.0581	......	.....	.....
1/4	.250	Drawn	0.0658	0.0600	0.181	0.167
17/64	.266	Drawn	.....	0.0680	.....	.....
9/32	.281	Drawn	.....	0.0758	.....	.....
19/64	.297	Drawn	.....	0.0847	.....	.....
5/16	.313	Drawn	0.103	0.0934	0.283	0.261
11/32	.344	Drawn	0.125	0.113	.....	.....
3/8	.375	Cold Finished	0.149	0.135	0.407	0.376
13/32	.406	Cold Finished	.....	0.158	.....	.....
7/16	.438	Cold Finished	0.203	0.184	0.555	0.511
15/32	.469	Cold Finished	.....	0.211	.....	.....
1/2	.500	Cold Finished	0.264	0.240	0.724	0.668
17/32	.531	Cold Finished	.....	0.271	.....	.....
9/16	.563	Cold Finished	0.335	0.304	0.917	0.845
19/32	.594	Cold Finished	.....	0.338	.....	.....
5/8	.625	Cold Finished	0.414	0.375	1.132	1.262
21/32	.656	Cold Finished	.....	0.413	.....	.....
11/16	.688	Cold Finished	0.501	0.454	1.369	1.262
3/4	.750	Cold Finished	0.595	0.540	1.630	1.502
25/32	.781	Cold Finished	.....	0.563	.....	.....
13/16	.813	Cold Finished	0.700	0.633	1.913	1.763
7/8	.875	Cold Finished	0.811	0.735	2.218	2.044
29/32	.906	Cold Finished	.....	0.788	.....	.....
15/16	.938	Cold Finished	0.930	0.844	2.546	2.347
1	1.000	Cold Finished	1.06	0.961	2.897	2.670
1 1/16	1.063	Cold Finished	1.20	1.08	3.271	3.015

TABLE 7—COMPARATIVE WEIGHTS OF STANDARD SIZES OF SCREW MACHINE STOCK[1]—Continued

11S-T3 and 17S-T ALLOYS[2]					BRASS AND STEEL (Round Wire and Rod)	
Diameter or Distance Across Flats—Inches		Standard Finish	Weight Lb. per Ft.[3]		Weight Lb. per Ft.	
Fraction	Decimal		Hexagonal	Round	Brass	Steel
1⅛	1.125	Cold Finished	1.34	1.22	3.667	3.380
1 5/32	1.156		....	1.28	.....	.....
1 3/16	1.188		1.49	1.35	4.085	3.766
1¼	1.250		1.65	1.50	4.527	4.172
1 5/16	1.313		1.82	1.66	4.991	4.600
1⅜	1.375		2.00	1.81	5.477	5.049
1 7/16	1.438		2.18	1.99	5.987	5.518
1½	1.500		2.38	2.16	6.519	6.008
1 9/16	1.563	Rolled	2.58	2.34	7.073	6.519
1⅝	1.625		2.80	2.54	7.650	7.051
1 11/16	1.688		....	2.74	8.250	7.604
1¾	1.750		3.25	2.98	8.873	8.178
1 13/16	1.813		....	3.15	9.518	8.773
1⅞	1.875		3.73	3.37	10.19	9.388
1 15/16	1.938		....	3.60	10.88	10.02
2	2.000		4.23	3.84	11.59	10.68
2 1/16	2.063		....	4.09	.....	.....
2⅛	2.125		....	4.34	13.08	12.06
2 3/16	2.188		....	4.59	.....	.....
2¼	2.250		....	4.86	14.67	13.52
2 5/16	2.313		....	5.13	.....	.....
2⅜	2.375		....	5.41	16.34	15.06
2 7/16	2.438		....	5.70	.....	.....
2½	2.500		....	6.00	18.11	16.69
2 9/16	2.563		....	6.30	.....	.....
2⅝	2.625		....	6.62	19.96	18.40
2 11/16	2.688		....	6.66	.....	.....
2¾	2.750		....	7.26	21.91	20.20
2⅞	2.875		....	7.94	23.95	22.07
3	3.000		....	8.64	26.07	24.03
3 1/16	3.063		....	9.00	.....	.....
3⅛	3.125		....	9.37	.....	.....
3¼	3.250		....	10.2	30.66	28.26

[1] Aluminum Alloy Screw Machine Stock is available in standard 12-foot lengths and in the finishes as listed above.

[2] For weight of 17S-T, multiply by 0.99. For weight of 2S, multiply by 0.96.

[3] Hexagonal stock is available only in the sizes for which weights are given.

TABLE 8—TOLERANCES

Diameter or Distance Between Parallel Faces Inches	Tolerance (Plus or Minus) Inches	
	Round	Hexagonal
1/8 to 1/2, incl.	0.0015	0.002
over 1/2 to 1, incl.	0.002	0.0025
over 1 to 1 1/2, incl.	0.0025	0.003
over 1 1/2 to 2, incl.	0.008	0.016
over 2 to 3, incl.	0.008	.....

SALES OFFICES

AKRON, O.	506 Akron Savings & Loan Building
ALBANY, N. Y.	90 State Street
ATLANTA, GA.	1818 Rhodes-Haverty Building
BOSTON, MASS.	20 Providence Street, Park Square
BUFFALO, N. Y.	1880 Elmwood Avenue
CHARLOTTE, N. C.	619 Johnston Building
CHICAGO, ILL.	520 North Michigan Ave.
CINCINNATI, OHIO	16th Floor, Times-Star Building
CLEVELAND, OHIO	1520 Midland Building
DALLAS, TEXAS	511 Tower Petroleum Building
DAVENPORT, IOWA	918 Kahl Building
DAYTON, OHIO	306 Harries Building
DENVER, COLO.	608 U. S. National Bank Building
DETROIT, MICH.	610 New Center Building
FAIRFIELD, CONN.	Boston Post Road
HARTFORD, CONN.	Capitol Building, 410 Asylum Street
INDIANAPOLIS, IND.	1008 Merchants Bank Building
KANSAS CITY, MO.	2306 Power & Light Building
LOS ANGELES, CALIF.	108 West Sixth Street
MILWAUKEE, WIS.	735 North Water Street
MINNEAPOLIS, MINN.	1060 Northwestern Bank Building
NEWARK, N. J.	2706 Newark and Essex Building
NEW YORK, N. Y.	230 Park Avenue
PHILADELPHIA, PA.	123 South Broad Street
PITTSBURGH, PA.	Local Sales, 1814 Oliver Building General Offices, Gulf Building
RICHMOND, VA.	213 Builders Exchange
ST. LOUIS, MO.	1002 Continental Building
SAN FRANCISCO, CALIF.	709 Rialto Building
SEATTLE, WASH.	1411 Fourth Avenue Building
SOUTH BEND, IND.	422 J.M.S. Building
TOLEDO, OHIO	1804 Ohio Building
WASHINGTON, D. C.	605 Southern Building
WICHITA, KAN.	411 4th National Bank Building

ALCOA ALUMINUM

The word "Alcoa" and the adjacent design are registered trademarks applied to the products of Aluminum Company of America, whose technical staff exercises the most rigid control over every process in the production of Alcoa Aluminum—from the mining of bauxite ore to the production of uniform and high quality aluminum and aluminum alloys, in every commercial form.

ALCOA ALUMINUM

Form No. AD-31B, 25-M-7-42; AD-31C, 35M-1-43

PRINTED IN U.S.A.
DAVIS & WARDE, IN
PITTSBURGH, PA